

# CRREL REPORT 87-18



US Army Corps  
of Engineers

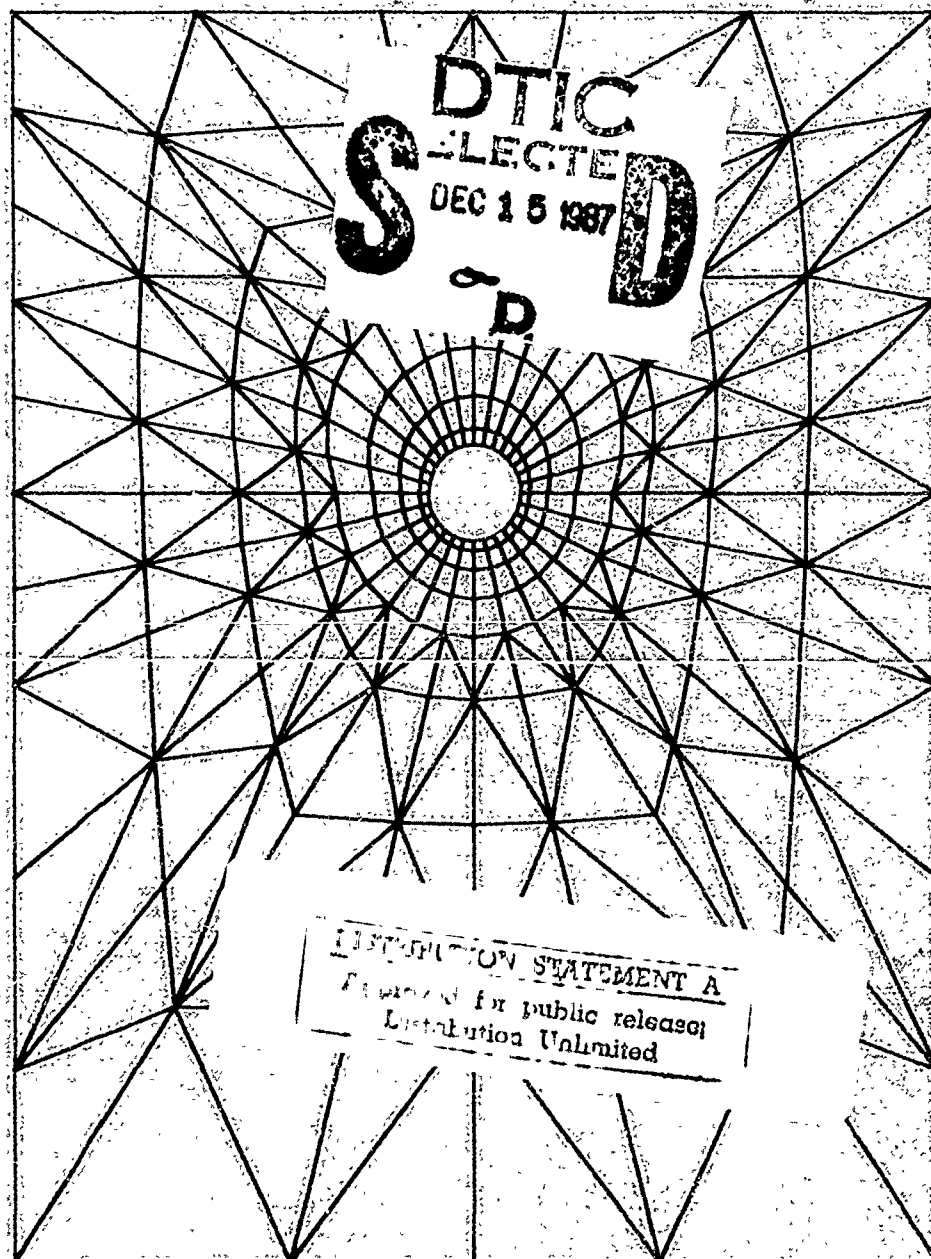
Cold Regions Research &  
Engineering Laboratory

4

AD-A186 939

DTIC FILE COPY

*An automatic finite element mesh generator*

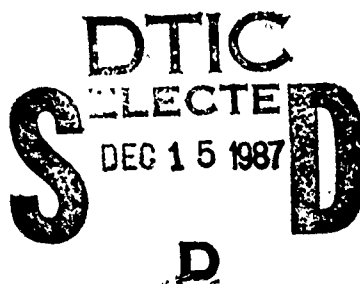


*Cover: Finite element mesh generated for  
modeling natural convection around a  
buried pipe.*

# CRREL Report 87-18

September 1987

(4)



## *An automatic finite element mesh generator*

Mary Remley Albert and James L. Warren

Accession For	
NTIS CRA&I	<input checked="checked" type="checkbox"/>
DTIC TAB	<input type="checkbox"/>
Unannounced	<input type="checkbox"/>
Justification	
By	
Distribution /	
Availability Codes	
Dist	Avail and/or Special
A-1	



## REPORT DOCUMENTATION PAGE

Form Approved  
OMB No. 07  
Exp. Date: J

1a. REPORT SECURITY CLASSIFICATION Unclassified			1b. RESTRICTIVE MARKINGS <b>AD-A186939</b>		
2a. SECURITY CLASSIFICATION AUTHORITY			3. DISTRIBUTION / AVAILABILITY OF REPORT Approved for public release: distribution is unlimited.		
2b. DECLASSIFICATION / DOWNGRADING SCHEDULE					
4. PERFORMING ORGANIZATION REPORT NUMBER(S) CRREL Report 87- 18			5. MONITORING ORGANIZATION REPORT NUMBER(S)		
6a. NAME OF PERFORMING ORGANIZATION U.S. Army Cold Regions Research and Engineering Laboratory		6b. OFFICE SYMBOL (If applicable) CECRL	7a. NAME OF MONITORING ORGANIZATION		
6c. ADDRESS (City, State, and ZIP Code) Hanover, New Hampshire 03755-1290			7b. ADDRESS (City, State, and ZIP Code)		
8a. NAME OF FUNDING / SPONSORING ORGANIZATION		8b. OFFICE SYMBOL (If applicable)	9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER		
8c. ADDRESS (City, State, and ZIP Code)			10. SOURCE OF FUNDING NUMBERS		
			PROGRAM ELEMENT NO.	PROJECT NO. 4A161 101A91D	TASK NO. 00
11. TITLE (Include Security Classification) An Automatic Finite Element Mesh Generator					
12. PERSONAL AUTHOR(S) Albert, Mary Remley and Warren, James L.					
13a. TYPE OF REPORT		13b. TIME COVERED FROM _____ TO _____		14. DATE OF REPORT (Year, Month, Day) September 1987	
15. PAGE COUNT 32					
16. SUPPLEMENTARY NOTATION					
17. COSATI CODES			18. SUBJECT TERMS (Continue on reverse if necessary and identify by block number)		
FIELD	GROUP	SUB-GROUP	Computer code; Fluid dynamics; Solid mechanics		
			Differential equations; Heat transfer;		
			Finite element; Mathematical methods		
19. ABSTRACT (Continue on reverse if necessary and identify by block number) Finite element computer codes are used in a variety of fields to solve partial differential equations of importance in science and engineering. The initial input to all of these programs requires the formation of a mesh (i.e., extensive lists of geometrical data listed in particular orders), and the success of the solution depends on a well-formed mesh. This report documents a mathematical mapping technique and its implementation into a user-friendly computer code that will automatically generate quality finite element meshes. This versatile generator uses standard FORTRAN, requires no special equipment (such as a computer terminal), is very economical to run, and is user-friendly. The mathematical technique is discussed, advantages and limitations of the method are presented, and examples are shown, and notes on user instructions are provided. <i>Keywords: /</i>					
20. DISTRIBUTION / AVAILABILITY OF ABSTRACT <input checked="" type="checkbox"/> UNCLASSIFIED/UNLIMITED <input type="checkbox"/> SAME AS RPT <input type="checkbox"/> DTIC USERS			21. ABSTRACT SECURITY CLASSIFICATION Unclassified		
22a. NAME OF RESPONSIBLE INDIVIDUAL Mary Remley Albert			22b. TELEPHONE (Include Area Code) 603-646-4422		22c. OFFICE SYMBOL CECRL-BA

## PREFACE

This report was prepared by Mary Remley Albert, Mechanical Engineer, Applied Research Branch, and James L. Warren, Mathematics Aide, Civil Engineering Research Branch, Experimental Engineering Division, U.S. Army Cold Regions Research and Engineering Laboratory. This work was sponsored by DA Project 4A161101A91D, *In-House Laboratory Independent Research*; Task 00, Work Unit 460, *Automated Mesh Generation*.

The authors wish to thank Kevin O'Neill and Larry Danyluk for their technical review, and Paul Richmond for being the test subject during the development stage of making the program user-friendly.

The contents of this report are not to be used for advertising or promotional purposes. Citation of brand names does not constitute an official endorsement or approval of the use of such commercial products.

## CONTENTS

	Page
Abstract.....	i
Preface .....	ii
Introduction.....	1
Mesh generation techniques .....	1
Transfinite mappings.....	2
Computer program, MESH.....	8
User instructions .....	9
Defining the boundaries of the mesh prior to running MESH.....	9
Generating a finite element mesh using MESH.....	11
Conclusions.....	15
Literature cited .....	15
Appendix A. Subroutines called by MESH.....	17
Appendix B. Example run using program MESH .....	21

## ILLUSTRATIONS

### Figure

1. An irregular region to be filled with a mesh .....	3
2. Unit square in $(u, v)$ space .....	3
3. Use of the bilinear projector to create a mesh for the region depicted in Figure 1 .....	4
4. Filling a region with highly distorted boundary curves with a mesh .....	4
5. Use of the trilinear projector .....	5
6. Overspill in a region with regular boundaries .....	5
7. Repositioning side nodes to eliminate overspill .....	6
8. Use of both the bilinear and trilinear projectors to create a mesh .....	6
9. Setting up a mesh using the bilinear projector for the area around a pipe.....	7
10. Triangular elements for the region around a buried pipe .....	7
11. Eight-noded elements on a curved surface .....	7
12. Use of the sine wave option .....	8
13. Use of the node numbering option .....	8
14. Example of merging four zones .....	8
15. Direction of numbering of the nodes on a side .....	10
16. An example of a side shared by two zones .....	10

## AN AUTOMATIC FINITE ELEMENT MESH GENERATOR

MARY REMLEY ALBERT AND JAMES L. WARREN

### INTRODUCTION

The finite element method is a well-established way for numerically solving partial differential equations of importance in science and engineering. It has been used with success in a wide variety of fields, including solid mechanics, heat transfer and fluid dynamics. However, a drawback of the method is that creating the initial input required for use of finite element computer programs can be a very time-consuming, tedious and error-prone procedure. The method requires the use of a mesh that is made up of nodes and elements. The initial input to finite element programs requires the construction of a list of all nodes and their coordinates, and a list of all elements and nodes that define the elements, listed in a particular order (known as the incidence list). Haber et al. (1981) estimate that roughly 80% of the total cost of a finite element analysis may be consumed by using conventional methods for the input.

In recent years, the development of automatic mesh generation techniques has relieved the analyst of the tiresome and costly chore of creating the coordinate and incidence lists. Several techniques have been presented, some based on heuristic methods, and others based on mathematical concepts.

It is the purpose of this paper to document the development and use of a user-friendly automatic mesh generator written by the authors and implemented in FORTRAN on the Prime computer at CRREL. The generator is based on the method of transfinite mappings. The advantages of the method will be discussed, some limitations of the method will be presented, and some notes on user instruction provided.

### MESH GENERATION TECHNIQUES

A variety of mesh generation techniques has been documented in the literature. Some of these are heuristic methods, which may work in specialized cases; others are based on mathematical concepts, and have a more predictable outcome. Of the latter type, three prominent methods are: 1) Laplacian schemes, 2) the use of isoparametric coordinates, and 3) the use of transfinite mappings. In this section, these three techniques will be briefly presented.

The first technique is generally termed the Laplacian scheme (Buell and Bush 1973). Essentially, this method involves the solution of Laplace's equation, where the variables represent coordinate locations in two-dimensional space:

$$\frac{\partial^2 x}{\partial u^2} + \frac{\partial^2 x}{\partial v^2} = 0 \quad (1)$$

$$\frac{\partial^2 y}{\partial u^2} + \frac{\partial^2 y}{\partial v^2} = 0. \quad (2)$$

It is expected that solution of diffusion equations for the coordinates will produce meshes where the nodes are distributed evenly on the interior of the grid. Equations 1 and 2 are then represented by a centered finite difference scheme. The finite difference equations are:

$$x_{i+1,j} + x_{i-1,j} + x_{i,j+1} + x_{i,j-1} - 4x_{ij} = 0$$

$$y_{i+1,j} + y_{i-1,j} + y_{i,j+1} + y_{i,j-1} - 4y_{ij} = 0.$$

Here,  $i$  increments nodes in the  $u$  direction, and  $j$  increments nodes in the  $v$  direction. These equations are then solved to find the coordinates ( $x_{i,j}$ ,  $y_{i,j}$ ) of each node in the interior of the grid. This method generates grids where the nodes are fairly evenly spaced on the interior of the grid, but this is not always a desired effect. There are variations on the method. For example, Hermann (1976) extends the method to more general grid types, and combines the use of the Laplacian scheme with isoparametric mappings. Denayer (1978) presents techniques for generating element connectivity and improving the computational efficiency of the method.

The other two methods, isoparametric and transfinite, rest on the premise that if the region to be filled with a mesh can be mapped onto a rectangular or triangular space, then that space may be filled with straight horizontal and vertical lines, yielding a checkerboard effect. A checkerboard requires only a simple programmed algorithm to generate a list of nodal coordinates and an incidence list. The methods then map the checkerboard pattern back onto the original region.

The use of isoparametric coordinates, as presented by Zienkiewicz and Phillips (1971), is an extension of the use of isoparametric coordinates for curvilinear finite element analysis. Shape functions are associated with each of the boundary nodes, and the interior coordinates are a linear combination of the product of each boundary node and its shape function. A major restriction on the method is that each side of the region must be smooth; slope discontinuities may occur only at the corners.

The method of transfinite mappings, discussed by Gordon and Hall (1973) and Habet et al. (1981), is more general than the use of isoparametric coordinates. One of its greatest advantages is that slope discontinuities may occur anywhere along any boundary. In the next section, this method is discussed in more detail.

## TRANSFINITE MAPPINGS

This method of mesh generation involves the mapping of a smooth and simple region onto the region of interest. Because the mapping guarantees that the boundary generated can match the true boundary at a non-denumerable number of points, the method is termed "transfinite."

Consider first a region defined by four sides, where the sides are not necessarily straight or smooth lines, as illustrated in Figure 1. The object of the mapping is to determine uniquely the location of interior nodes, given the location of nodes on the sides of the region. To do this, the unit square will be mapped onto the four-sided region.



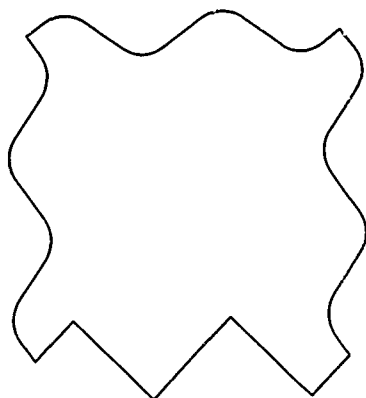


Figure 1. An irregular region to be filled with a mesh.

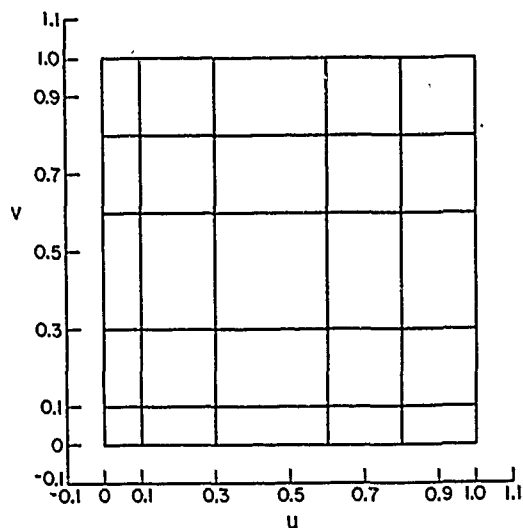


Figure 2. Unit square in (u,v) space.

The unit square is described in a coordinate system in  $u$  and  $v$  such that  $0 \leq u \leq 1$ ,  $0 \leq v \leq 1$ , as illustrated in Figure 2. By assuming that nodes on opposite sides of the square have the same  $u$  or  $v$  values, it is a trivial matter to generate coordinate and incidence lists for the square by assigning the location of nodes on the interior of the square to points of intersection of lines of constant  $u$  or  $v$ . When the above process is complete, this system of subdivision may be mapped to the irregular, four-sided region. This transfinite mapping is referred to in the literature (Gordon and Hall 1973, Haber et al. 1981) as the "bilinear projector," and is defined as follows.

$$\begin{aligned}
 P_B(u,v) = & (1-v) \psi_1(u) + v\psi_2(u) + (1-u) \xi_1(v) + u\xi_2(v) \\
 & - (1-u)(1-v) F(0,0) - (1-u) vF(0,1) \\
 & - uvF(1,1) - u(1-v) F(1,0)
 \end{aligned}$$

where

$$0 \leq u \leq 1, \quad 0 \leq v \leq 1.$$

In eq 3,  $\psi_1(u)$ ,  $\psi_2(u)$ ,  $\xi_1(v)$  and  $\xi_2(v)$  represent the curves that describe the location of the edges in  $x,y$  space. The coordinates of the corners are indicated by  $F(u,v)$ . A mesh generated to fill the region depicted in Figure 1 is illustrated in Figure 3. Note that the mesh on the interior of the region reflects the boundary shapes. It is evident that, to generate a mesh for use with finite elements,  $\psi$  and  $\xi$  are not required to be smooth functions, or even any known functions. The mapping requires only the coordinate values at the nodal points along the sides. Slope discontinuities may occur anywhere along any boundary.

If the region to be filled with a mesh has highly distorted boundary curves, the interior mesh generated may lie outside the boundary. This was noted by Gordon and Hall (1973) and Haber et al. (1981) for transfinite mappings. Note that this problem also occurred in the use of isoparametric mappings (Hermann

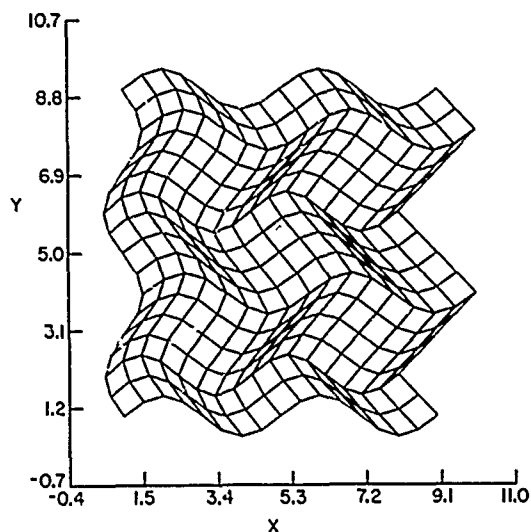
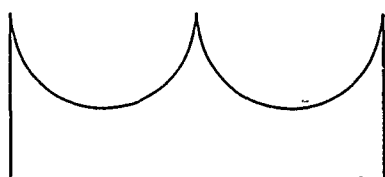
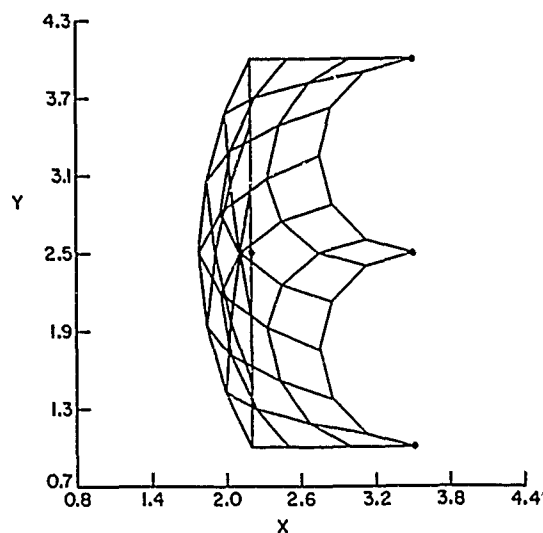


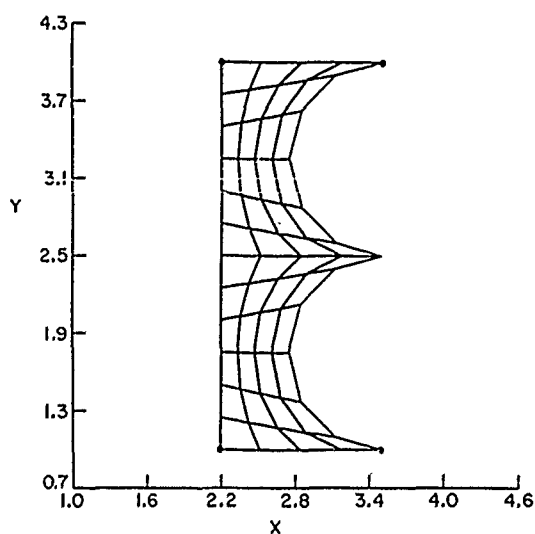
Figure 3. Use of the bilinear projector to create a mesh for the region depicted in Figure 1.



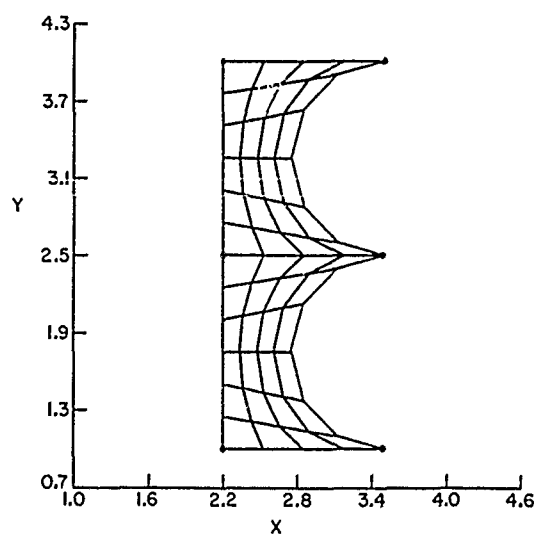
a. Region to be filled with a mesh.



b. Overspill with the chosen corners.



c. Redefining the corners to eliminate overshoot.



d. Subdividing the region to eliminate overshoot.

Figure 4. Filling a region with highly distorted boundary curves with a mesh.

1976). As an illustration, consider the region outlined in Figure 4a. When the bilinear projector is used, and the four corners are as indicated in Figure 4b, the resultant mesh contains overspill. In this case, two remedies are available. First, by redefining the corner locations to those shown in Figure 4c, the overspill is eliminated, as illustrated in that figure. The overspill may also be eliminated by dividing the region into simpler subregions, which may then be filled with meshes separately and merged together, as illustrated in Figure 4d.

Haber et al. (1981) present the "trilinear projector" that may be used in creating meshes for three-sided regions. Again, slope discontinuities may occur along the boundaries. The trilinear projector is defined as follows.

$$\begin{aligned}
 P_T(u,v,w) = & \frac{1}{2} \left[ \left( \frac{u}{1-v} \right) \xi(v) + \left( \frac{w}{1-v} \right) \eta(1-v) + \left( \frac{v}{1-w} \right) \eta(w) \right. \\
 & + \left( \frac{u}{1-w} \right) \psi(1-w) + \left( \frac{w}{1-u} \right) \psi(u) + \left( \frac{v}{1-u} \right) \xi(1-u) \\
 & \left. - w\psi(o) - u\xi(o) - v\eta(o) \right] \\
 & 0 \leq u \leq 1, \quad 0 \leq v \leq 1, \quad 0 \leq w \leq 1, \quad u + v + w = 1.
 \end{aligned} \tag{4}$$

The use of this projector is illustrated in Figure 5. The coordinates in the transformed space,  $u$ ,  $v$ , and  $w$ , must lie between 0 and 1, must sum to 1, and run counterclockwise along the edges around the three-sided region.

Albert (1984) observed that overspill may occur also in regions with very regular boundaries, but where the nodes along the edges are highly bunched. An example of overspill in this situation is depicted in Figure 6. In Figure 7, by rearrangement of the position of nodes along one of the edges, we see that the situation has been corrected.

It may be observed that the bilinear projector produces meshes with four-sided elements, and the trilinear projector produces meshes with three-

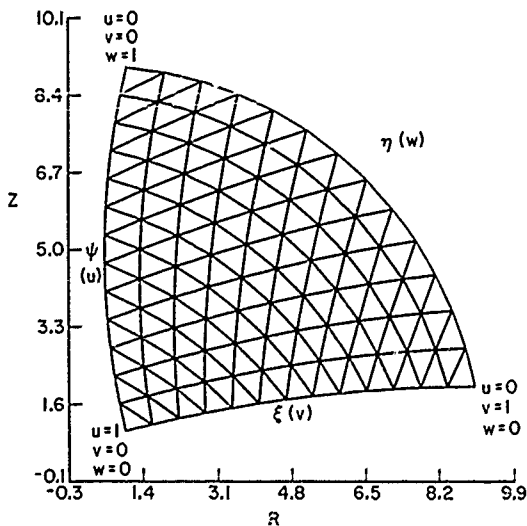


Figure 5. Use of the trilinear projector.

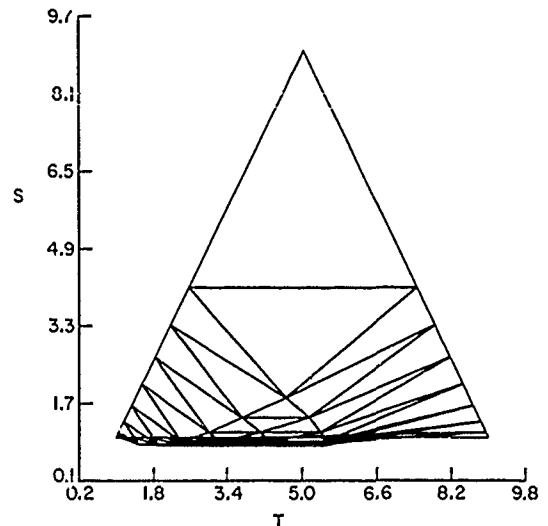


Figure 6. Overspill in a region with regular boundaries.

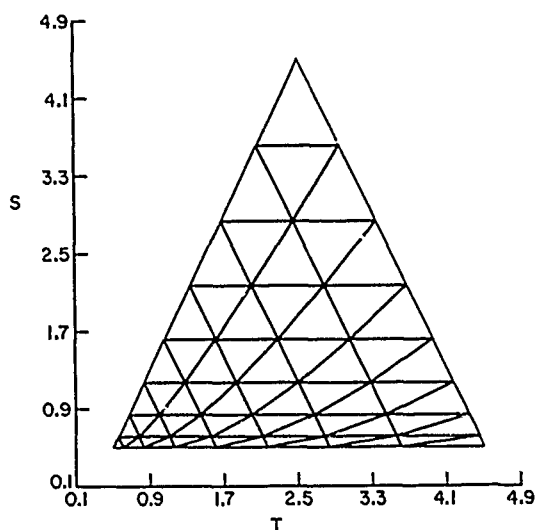
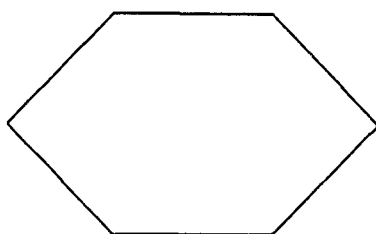
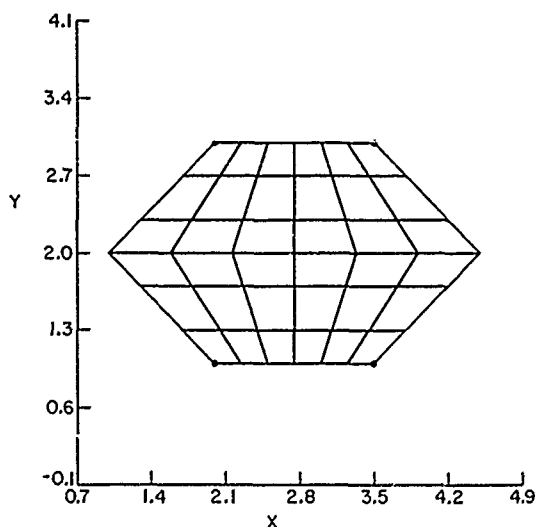


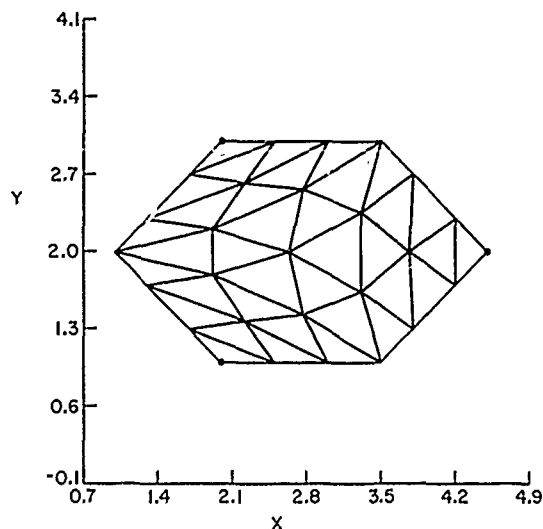
Figure 7. Repositioning side nodes to eliminate overspill.



a. Region to be filled with a mesh.



b. Defining the region as a four-sided region.



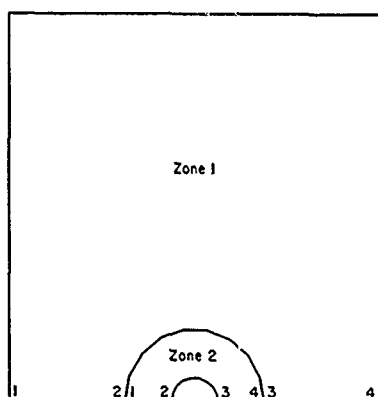
c. Defining the region as a three-sided region.

Figure 8. Use of both the bilinear and trilinear projectors to create a mesh.

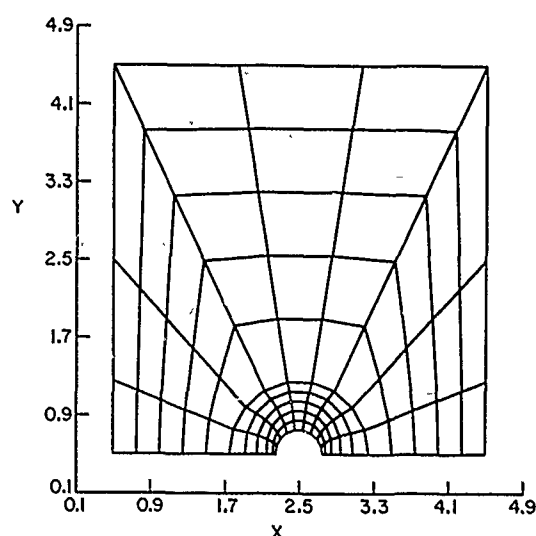
sided elements. In program MESH, presented here, the user has an option to have diagonals drawn through the four-sided elements generated from the bilinear projector to create triangular elements. Also, in MESH, the user has an option to generate eight-noded quadrilaterals by removing the center node from a cluster of four-noded quadrilaterals from the bilinear projector. Use of the trilinear projector will always result in triangular elements in MESH.

Now let us observe some of the properties of these mappings. First, the possibility of allowing slope discontinuities along any side means that traditional concepts about the word "corner" may be abandoned. For example, Figure 8a outlines a region to be filled with a mesh. In Figure 8b, the bilinear projector is used to create the mesh, and in Figure 8c, the trilinear projector is used. In both cases, the "corners" of the region are marked with a solid dot.

Figure 9 illustrates a combination of two zones created using



a. Two zones for the region around a pipe.



b. Use of the bilinear projector with four-noded quadrilateral elements in the two zones depicted in Figure 9a.

Figure 9. Setting up a mesh using the bilinear projector for the area around a pipe.

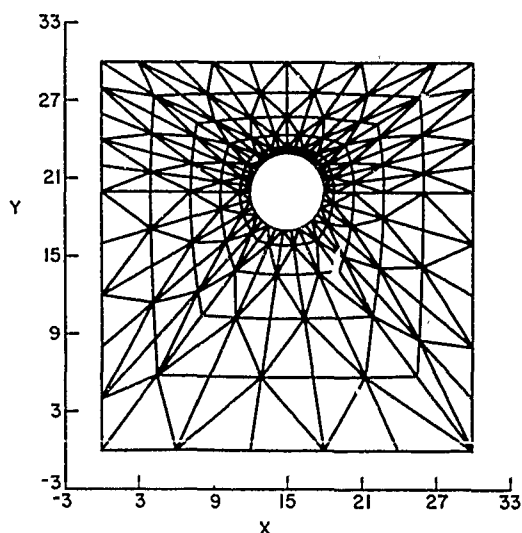


Figure 10. Triangular elements for the region around a buried pipe.

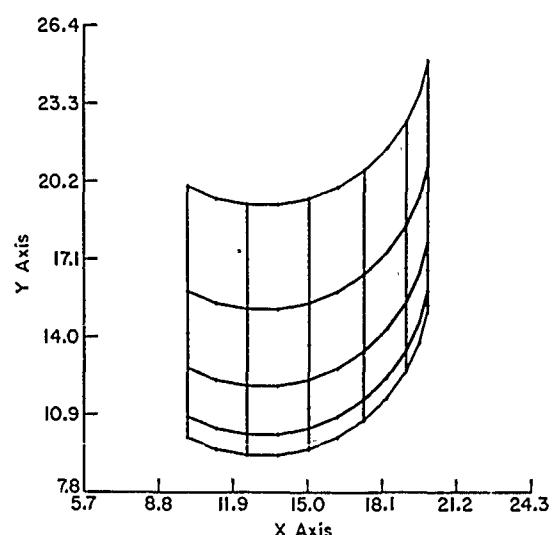


Figure 11. Eight-noded elements on a curved surface.

the bilinear projector to set up a mesh for a possible application involving the area around a pipe. In program MESH, a variety of options are made available to the user. Figure 10 depicts the use of triangular elements to model the complete region around a buried pipe. Figure 11 depicts the use of eight-noded elements. In Figure 12, the use of a sine wave generator for automatically inputting edge nodes is displayed. The program also has an option to allow the nodes to be numbered, as depicted in Figure 13. In Figure 14, a more complex figure is represented by merging four zones.

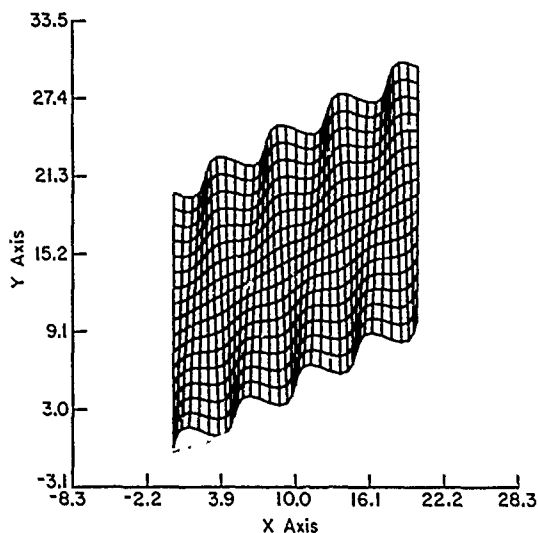


Figure 12. Use of the sine wave option.

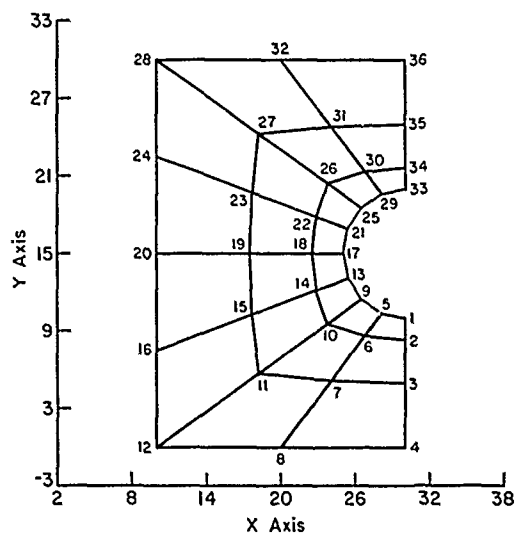


Figure 13. Use of the node numbering option.

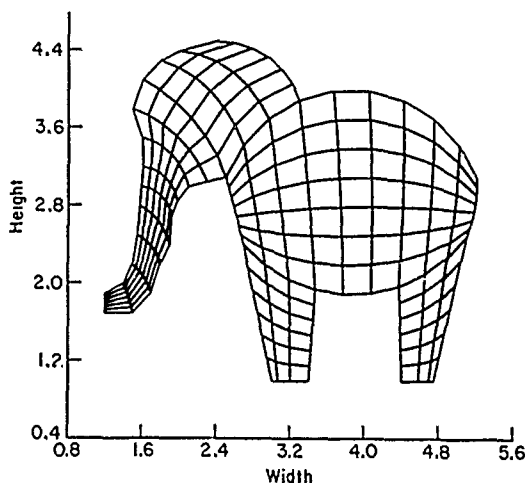


Figure 14. Example of merging zones: a finite elephant.

## COMPUTER PROGRAM, MESH

This program is written in FORTRAN 77 and implemented on the Prime computer at CRREL. At the time of publication of this report, all interaction between the analyst and computer occurs at the computer terminal. The program does not require the use of a special terminal, and a digitizer is not required. The plotting program is designed for use with an HP 7580A plotter.

Note that throughout this section, and the rest of the report, the use of the word "zone" denotes one region that is filled with a mesh by use of either the bilinear or trilinear projectors. A complicated region may be subdivided into zones, meshes may be

created for each zone separately, and the zones may then be joined together, or merged, to create incidence and coordinate lists with one continuous global numbering.

MESH is the main program. MESH calls subroutines ZONE, RATIO, TRFIN, TRQUAD, TRTRI, RENMBR, MERGE, MESHPLLOT and BANDWIRED. It displays the MESH main menu and performs virtually all the tasks that a user would want except actual mesh construction. From the main menu the user can 1) specify the number of zones in the mesh, 2) call the ZONE subroutine, where zone construction actually occurs, 3) call the MESHPLLOT subroutine, which plots any previously constructed mesh, 4) change filename for the next mesh, 5) get a mesh from a file (gets all the zone's material type, element type and boundary node information), 6) save a mesh to a file (saves all the zone's material type, element type and boundary node information), 7) call the subroutine named BANDWIRED, which reduces the bandwidth of a mesh

and 8) exit the program and return to PRIMOS (Prime Operating System). There are 25 subroutines called by MESH; these subroutines are briefly described in Appendix A.

The output of program MESH consists of several files, which contain incidence and coordinate lists, plotting information, and general and edge information on the last finite element mesh that was created. The user provides the general filename during the run of the program; this name is referred to as "filename" in the discussion of output files that follows.

File "filename.coor" contains the coordinate list for the mesh. The first line contains a number that indicates the number of nodes in the entire mesh. This is followed by a list of the node numbers and corresponding coordinates.

File "filename.incd" contains the incidence list for the mesh. The first line contains two numbers that indicate the number of elements in the entire mesh, and the number of nodes per element. This is followed by a list of the element numbers and corresponding nodes for each element (listed counterclockwise), and the material type for each element.

File "filename.plot" is required if program MESH is used to plot the mesh. The file contains the number of zones in the mesh, the plotter pen color for each zone, the first element number of each zone, and the element type number for each zone.

File "filename.edge" will not exist unless the user requests that the information be saved by using option 6 on the main menu. This file is required if a mesh was only partially completed, or if the user wishes to change part of the mesh at a later time. This file contains the number of zones in the mesh, the number of sides in the zone, the number of nodes on sides 1 and 3 and 2 and 4 for each zone, the material type number for each zone, the element type number for each zone, and the coordinates of each corner followed by the coordinates of the nodes on that side (i.e., the coordinates of corner 1 with the coordinates of the nodes on side 1, etc.).

File "meshinfo" contains useful information for use in running program MESH after the first time. It contains the number of zones in the last mesh created, the number 1 or 2 to indicate whether the last mesh's edge information was or was not saved, and the filename of the last mesh created.

## USER INSTRUCTIONS

Although the reader who wishes to generate a mesh may be tempted to skip the first part of this report, and begin his or her investigation by reading this section, it is our firm belief that no computer program should be used completely as a "black box." The potential user should grasp a basic understanding of the method and brief introduction to some of the benefits and pitfalls before sitting down at the computer terminal. This information is provided in the first four sections of the report. Therefore, before using program MESH, you are urged to read this brief report in its entirety.

### Defining the boundaries of the mesh prior to running MESH

You will make the best use of your time (and computer time) if you make some basic decisions before getting onto the computer. This planning stage requires pencil, paper and some thoughtful reflection on your part.

First, construct and label the coordinates on the x and y axes. Sketch the outline of the region to be filled with a mesh. If the region has more than four sides, or if it is a very contorted region, or if the region is composed of

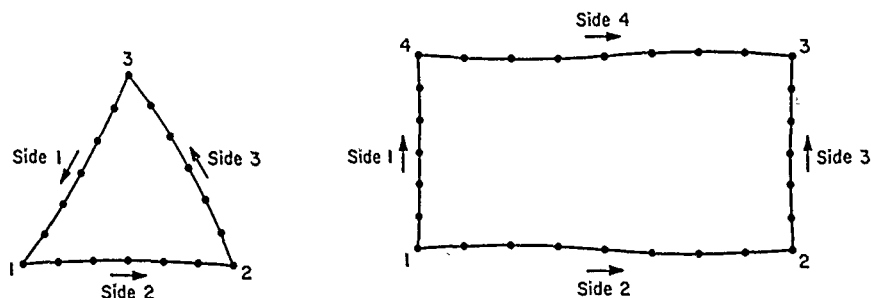


Figure 15. Direction of numbering of the nodes on a side.

more than one material, it may be desirable to divide the region into smaller subregions, called "zones." Each zone may have either three or four sides. (Note that for the mapping, the word "side" is very general, and may include a line segment with a  $90^\circ$  angle, for example. This concept is discussed in earlier sections of the report.) Number each zone, starting with 1. Assign to each zone a material type; the material type is an integer that will be used to assign material properties (such as density, conductivity, etc.) to the elements for the finite element program.

For each zone, number each of the corners, starting with 1, in a counter-clockwise direction. We suggest that the lower left-hand corner of each zone be designated as corner 1 for that zone. For each zone you must decide if you want triangular elements, four-noded quadrilateral elements, or eight-noded quadrilateral elements.

For each zone, number the sides, starting with 1, in a counterclockwise direction. The convention for the numbering is important: for a three-sided region, side 1 connects corners 1 and 3, and for a four-sided region, side 1 connects corners 1 and 4. We suggest that you write down information on corners and sides for reference when you are running the program. Also, the mesh generation technique used—transfinite mappings—relies on a specific direction of numbering for the nodes on a side (not always a counterclockwise direction). Thus, it will be important for you to be aware of the direction of each side, especially for

sides that two zones have in common. The convention for this is illustrated in Figure 15.

Check to be sure that a side that is shared by two zones will run in a direction consistent with the convention for each zone. As an example, Figure 16 depicts the directions for each side for the mesh illustrated in Figure 9b. Note that side 3 of zone 1 and side 1 of zone 2 are shared sides, yet the direction is compatible with the convention for each zone. (Note that it is possible to match sides whose nodes run in different directions, but in that case the nodes will have to be matched one by one.)

Determine how the edge nodes

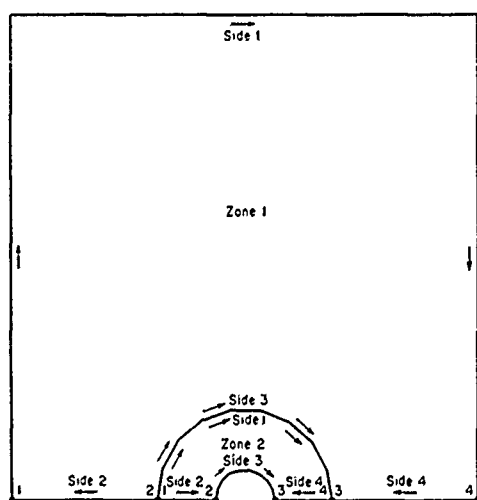


Figure 16. An example of a side shared by two zones.



on each side of each zone are to be constructed, as follows. First, decide how many nodes are to be on each side, remembering that a three-sided region must have the same number of nodes on each side, while in a four-sided region, alternate sides must have the same number of nodes. Also, if eight-noded quadrilateral elements are to be used in a four-sided region, there must be an odd number of nodes on each side. Next, decide how the nodes should be spaced along each side. The following options\* are available for specifying the spacing of the nodes along each side:

1. You specify the nodes.
2. Interpolation along a straight line.
3. Interpolation along a circle.
4. Interpolation along a sine wave.

The first option allows you to type in the coordinates of each node on the side. If this option is to be selected, you should make a list now of the nodal coordinates, making sure that they are specified in the proper direction for that side, as discussed above. For the remaining options, MESH automatically computes the coordinates of the edge nodes. The linear interpolation and sine wave options allow for the following node spacings along the side:

1. Evenly spaced nodes.
2. Spacing as the square root of distance.
3. Linearly increasing spacing.
4. Linearly decreasing spacing.
5. A more severe case 3.
6. A more severe case 4.

If option 5 or 6 is chosen, you are given a choice of how clustered the nodes are to be along the line by specifying the fraction of the line that is to be occupied by the first element. The remaining line segment is then divided up by the program in either a linearly increasing or linearly decreasing fashion, as indicated.

After you have determined the shape of the mesh and decided on the details of the sides and corners of each zone, it is time to go to the computer and execute the program MESH. MESH does not require the use of a special terminal or digitizer; it was designed to run interactively on any terminal. The next section describes the procedure.

### **Generating a finite element mesh using MESH**

The mesh-generating program, MESH, is menu-driven, affording the user much flexibility in the procedure. In this section, each menu will be presented, with a brief description of the options in each menu.

#### *Main menu*

When the program is run, the first menu to appear is the main menu (shown on the following page). From this menu you can initiate the formation of a new mesh (options 1,2), retrieve a previously formed mesh (option 5); work on a mesh (options 4,6,7), plot the mesh (option 3), or exit to the operating system (option 8).

If you wish to create a new mesh, proceed by selecting option 1, followed by option 2. The second option causes the zone menu to appear, which allows the user to perform all of the operations necessary for the creation of a mesh in each zone. Option 3, "Plot a mesh," is used for plotting the mesh

\* Shown exactly as they are produced by MESH.

## MESH MAIN MENU

1. Specify # of zones in the mesh. Currently not yet defined.
2. Input the zones.
3. Plot a mesh.
4. Change the name of the mesh. Currently: "SAMPLE "
5. Get a mesh from a file.
6. Save the information on the edges of the mesh "SAMPLE "
7. Reduce the bandwidth of a mesh.
8. Exit to Primos Operation System.

on the HP 7580A plotter. Normally, each material in the mesh is plotted in a different color. Also, the mesh may be plotted with or without node numbers. The plot option may be called either if a mesh has just been created by MESH, or if the incidence, coordinate and plotfile lists for a mesh already exist, i.e., if MESH has been run previously.

The fourth option in the main menu, "Change the name of the mesh," is used to specify the names of the incidence, coordinate and plot, and edge information files of the mesh to be generated by this run of the program. The default mesh name is that for the previous mesh; this name is listed in the option. The resulting incidence and coordinate lists will have the filename as specified by option 4 with ".incd" and ".coor" extensions on the name. A file needed by the plotting subroutine will have the same filename with a ".plot" extension. If you opt to save the information on edges and corners of the mesh, that information will be put into a file with filename (as specified in option 4) with an ".edge" extension.

Option 6, "Save the information on the edges of the mesh," is used when a mesh has been input and you wish to save the zone's edge conditions. This is useful for editing the mesh after it has been created; you may alter parts of the existing mesh (change the shape of a side of a zone or change node spacing, for example) without having to input the entire mesh again. Option 6 also allows you to save the information on the sides and edges of a zone after each zone has been completed. This is useful if you are generating a multi-zone mesh but are called away from the computer before completing the entire mesh. Option 5, "Get a mesh from a file," allows the edge conditions for a previously formed mesh (saved using option 6) to be read in.

Option 7, "Reduce the bandwidth of a mesh," examines the coordinate and incidence lists for a mesh, and renumbers the nodes to reduce the bandwidth of the mesh. This option should be the very last operation performed on a mesh once it has been generated. Then, selecting option 8 closes all files and exits the program to PRIMOS.

### *Zone menu*

The zone menu appears when you have selected option 2 from the main menu, "Input the zones." The appearance of this menu changes slightly throughout the mesh generation procedure to provide you with useful information. The first time that it is called, the zone menu appears as seen on the top of the next page.

If option 6 of the main menu ("Get a mesh from a file") was used to read in all of the zone's edge conditions, then select option 10 of the zone menu. Either the zone menu will reappear, with the current zone number being one higher than the last zone number, or the program will return to the main program if the last zone number equals the number of zones in the current mesh. If the mesh was not read from a file, you should begin by selecting

1. Specify the # of sides. The # of sides = not yet defined.
2. Input the corners.
3. Input the # of nodes. The # of nodes = not yet defined.
4. Input the option for generating the nodes along any side.
5. Set the material of zone. The material # = not yet defined.
6. Set the element type. The element type is not yet defined.
7. Help on the zone construction.
8. Change, insert, delete, or match nodes.
9. Recalculate the edge nodes with previous side options.
10. File Zone # 1 and go to next the Zone or the Main Menu.
11. Emergency exit to the Main Menu.

option 1 to enter the number of sides in the current zone (either 3 or 4). The program will either prompt you for the corners, or if the current zone number is greater than one, then the program will ask if the user wants "automatic matching" to occur.

Automatic matching means that an entire or partial side of a previously entered zone is mapped onto an entire side of the current zone, thus this defines not only the corners of the side but also the number of nodes on the side and the coordinates of all the nodes on the side. (If you do not wish to map up an entire side of the current zone, you should choose option 8 on the zone menu later, after all of the corners and nodes on all the sides are defined.) If automatic matching is selected, the program will ask for the number of the zone from which you are matching, the number of the side from which matching will occur, and the number of the side on the current zone that will be matched onto. You also must input whether or not the entire side of the previous zone would be matched up, or whether the side should be matched up between specified nodes on the previous zone's side.

Option 3 of the zone menu, "Input the # of nodes," should be selected to specify the number of nodes on each side of the zone. The use of transfinite mappings requires that three-sided zones have the same number of nodes on each side, while for four-sided zones, alternate sides must have the same number of nodes. Also, if eight-noded quadrilateral elements are to be used, then there should be an odd number of nodes on each side.

The fourth option allows you to select the spacing of nodes along each side. It can be used to set the side option for the first time, or it can be used to change an existing side option. The options offered are:

1. You specify the nodes.
2. Interpolation along a straight line.
3. Interpolation along a circle.
4. Interpolation along a sine wave.

If option 1 is selected, you will be prompted to type in the coordinates of each node along the side. If interpolation along a straight line or sine wave is selected, you will be given the following choice of spacings:

1. Evenly spaced nodes.
2. Spacing as the square root of distance.
3. Linearly increasing spacing.
4. Linearly decreasing spacing.
5. A more severe case 3.
6. A more severe case 4.

An explanation of these options was given in the previous section.

Option 5 for the material type takes an integer as input. This integer may be used later in the finite element code to indicate material properties. For MESH, this integer may correspond to a pen color when the mesh is plotted.

The sixth option allows for the specification of element type. If the zone is three-sided, a message will appear saying that three-sided zones must have triangular elements. A four-sided zone may contain elements that are triangles, four-noded rectangles, or eight-noded rectangles.

Option 8 allows you to change the coordinates of an existing side node and insert, delete or match nodes along a side. In this case the matching allows you to match a partial side of the current zone onto some or all of the nodes specified in a previous zone. This includes the special case where two zones may share only one node. If this option is selected, the node will be displayed as follows:

Node Menu - Currently Zone #1

1. Change the coordinates of nodes.
2. Insert nodes.
3. Delete nodes.
4. Match nodes from other zones onto nodes of this zone.
5. Return to the zone menu.

For any of these options, the program displays the node numbers and coordinates as they currently exist, then prompts the user as to what must be done to accomplish the changes.

Option 9 is used to recalculate sides with options previously entered. This option is most useful when an entire mesh has been entered and plotted, but some of the zones in the plot have too many or not enough nodes on a side, for example. If this situation develops, you should again select option 2 ("Input the zones") on the main menu. Then, when the zone menu comes up, select option 3 ("Change # of nodes") and specify the number of nodes desired. Then select option 9 to recalculate all of the edge nodes of all the sides, using the previous side options but the new number of nodes per side. Option 9 may not be used if the mesh has been read in from a file. When all of the nodes on all of the sides have been specified as desired, select option 10.

Option 11 allows you to leave the zone menu and go to the main menu to save the information collected so far on completed zones. This is useful if you are called away from the terminal before the mesh is complete. After selecting option 11 of the zone menu, you should select option 6 of the main menu to save the information.

As the information in the zone menu is specified, the appearance of the menu changes to indicate the information that currently exists. For example, after options 1, 2, 3 and 6 have been selected, the zone menu may appear as shown at the top of page 15. The program has many checks to try to ensure that the information supplied by the user is complete and in accordance with the theory behind the mapping used.

When all of the zones have been created, the program will ask you to supply brief information on merging (joining) the zones. Every two zones that share nodes must be merged, in order that the shared nodes have only one node number, and global coordinate and incidence lists can be generated. Note that it is your responsibility to check the resulting mesh to be sure

ZONE MENU Current zone # 1

1. Specify the # of sides. The # of sides = 4.
2. Change the corners.

	X	Y
Corner 1 is	1.00000	1.00000
Corner 2 is	4.50000	1.00000
Corner 3 is	6.00000	3.00000
Corner 4 is	2.50000	8.50000

3. Change the # of nodes.  
There are 6 nodes on side 1, and 7 nodes on side 2, currently.
4. Input the option for generating the nodes along any side.
5. Set the material of zone. The material # = not yet defined.
6. Change the element type. The element type is 4 noded quadrilateral.
7. Help on the zone construction.
8. Change, insert, delete, or match nodes.
9. Recalculate the edge nodes with previous side options.
10. File Zone # 1 and go to the next Zone or Main Menu.
11. Emergency exit to the Main Menu.

that the edge nodes of the zones match. That is, a node that defines part of the side of one zone should not fall, for example, somewhere along the side of an element where no node is intended in the second zone. Also, the program does not assume that the complete side of one zone matches the complete side of another zone; rather, zones are merged on a node-by-node comparison. This allows a more general overall mesh formation.

## CONCLUSIONS

A program has been developed using the method of transfinite mappings to automatically generate finite element meshes, given the location of nodes on the boundary of the mesh. The program is easy to use, and may generate meshes for regions involving complicated shapes. Advantages and disadvantages of the method, and a variety of examples have been presented.

## LITERATURE CITED

- Albert, M.R. (1984) Modeling two-dimensional freezing using transfinite mappings and a moving mesh finite element technique. USA Cold Regions Research and Engineering Laboratory, CRREL Report 84-10.
- Buel, W.R and B.A. Bush (1973) Mesh generation—a survey. *Journal of Engineers in Industry, Transactions of the American Society of Mechanical Engineers*, Ser. B, 95: 1-15.
- Collins, R.J. (1973) Bandwidth reduction by automatic renumbering. *International Journal for Numerical Methods in Engineering*, 6: 345-356.
- Denayer, A. (1978) Automatic generation of finite element meshes. *Computer Structures*, 9: 359-364.
- Fellers, G.E. (1984) HP plotter library subroutines. USA Cold Regions Research and Engineering Laboratory, Technical Note (unpublished).

- Gordon, W.J. and C.A. Hall (1973)** Construction of curvilinear coordinate systems and application to mesh generation. *International Journal for Numerical Methods in Engineering*, 7: 461-477.
- Haber, R., M.S. Shepard, J.F. Abel, R.H. Gallagher and D.P. Greenberg (1981)** A general, two-dimensional, graphical finite element preprocessor utilizing discrete transfinite mappings. *International Journal for Numerical Methods in Engineering*, 17: 1015-1044.
- Herrmann, L.R. (1976)** Laplacian-isoparametric grid generation scheme. *Journal of the Engineering Mechanics Division, American Society of Civil Engineers*, 102: 749-756.
- Sullivan, J.M., Jr. (1985)** LABLIB bandwidth reduction routines. LABLIB Numerical Methods Laboratory, Dartmouth College, Hanover, New Hampshire.
- Zienkiewicz, O.C. and D.V. Phillips (1971)** An automatic mesh generation scheme for plane and curved surfaces by isoparametric coordinates. *International Journal of Numerical Methods in Engineering*, 3: 519-528.

## APPENDIX A: SUBROUTINES CALLED BY MESH

### ZONE

ZONE is called by the main program. ZONE calls subroutines SPEC, LINR, CIRC, CALSIN, MAZONE, CHINDEL and WHICH. ZONE allows the user to input the information necessary for generating the mesh in a zone. The zone menu allows the user to 1) specify the number of sides, 2) define the coordinates of the corners, 3) set the number of nodes on the sides, 4) select the option for generating the nodes along any given side, 5) set the material type of the zone, 6) set the element type for the zone, 7) get help on how to construct zones, 8) change, insert, delete or match nodes, 9) recalculate the edge nodes of all the sides with the last used side options, and 10) file the current zone and go to the next zone or, if the current zone is the last zone, return to the main menu. When defining the coordinates of the corners, if the zone being constructed is not the first zone, then the subroutine will ask if the user would like to match up any entire sides of the current zone from previously constructed zones. If any mistakes are made the user can simply reselect anything desired and then input the corrected data. The four options for defining the coordinates of the nodes on any side of the zone are 1) specifying the nodes by hand, 2) linear interpolation along the side, 3) circular interpolation along the side, and 4) sine wave interpolation along the side. When the nodes on all the sides are specified correctly, ZONE lists the coordinates in a file named NODES, which is used by the mesh generation subroutines.

### MESHPLOT

MESHPLOT is called by the main program. MESHPLOT calls subroutines PLOTS, PLOT, NEWPEN, SYMBOL, AXIS, and NUMBER. All of the subroutines called by MESHPLOT are fully documented (Fellers 1984). MESHPLOT allows the user to plot a mesh. Given that the user has a mesh that has been constructed, the user can have it plotted with node numbers in the plot, or without node numbers. The user can have all the zones in the mesh plotted in black (pen color 1) regardless of each zone's material type, or can have each zone plotted in the color that corresponds to its material type. The user must specify the length of the x and y axis. The subroutine will scale the plot to the lengths desired and make sure that all the data will be on the plot. The user will be given the option to specify captions for the axes, and will be given the option of specifying a title for the plot.

### BANDWIRED

BANDWIRED is called by the main program. BANDWIRED calls only the subroutine BWRSUB. BANDWIRED reads in the incidence and coordinate lists of a mesh and then reduces the bandwidth of the mesh by calling BWRSUB. It then writes the new incidence and coordinates lists into the same files.

### BWRSUB

BWRSUB is called by subroutine BANDWIRED. It takes node and element arrays and reduces the bandwidth of the arrays using Collins' (1973) bandwidth reduction algorithm, as programmed by Sullivan (1985). It checks to make sure that all the nodes are assigned to an element. It returns the old and new half bandwidths.

## **RATIO**

RATIO is called by the main program. It calls subroutine WHICH. RATIO calculates the fraction of the total length from the corner of each side to each node on that side. The information is stored in an array named RAT(,), and is written in a file named RATIOs.

## **TRFIN**

TRFIN is called by the main program. TRFIN calls subroutine TRIANG. TRFIN performs the bilinear projector transfinite mapping to produce the coordinates of the nodes on the interior of the zone, given the edge coordinates. It also produces the incidence list for the zone by numbering the elements in a row-wise pattern. The elements produced are four-noded quadrilaterals.

## **TRIANG**

TRIANG is called by subroutine TRFIN. TRIANG triangularizes rectangular elements. It also produces the incidence list for the zone by numbering the elements in a pattern.

## **TRQUAD**

TRQUAD is called by the main program. TRQUAD performs the bilinear projector transfinite mapping to produce the coordinates of the nodes on the interior of the zone, given the edge coordinates. It also produces the incidence list for the zone by numbering the elements in a row-wise pattern. The elements produced are eight-noded quadrilaterals.

## **TRTRI**

TRTRI is called by the main program. TRTRI performs the trilinear projector to produce the interior node coordinates for a three-sided zone. By numbering the elements in a row-wise pattern, it also produces the incidence list for the zone.

## **RENMBR**

RENMBR is called by the main program. RENMBR rennumbers the nodes and elements of a zone. The elements are numbered starting from where the previous zone's element numbers ended. The node numbers also start from where the previous zone's node numbers ended.

## **MERGE**

MERGE is called by the main program. MERGE interactively asks the user which zones have nodes in common. It then does a search to find the nodes shared by two or more zones, and rennumbers the nodes so that each node in the problem has only one node number, its location is stored only once, and the coordinate and incidence lists are changed to be consistent with these changes.

## **MAZONE**

MAZONE is called by subroutine ZONE. MAZONE matches up sides of zones with previously entered sides of earlier zones. It must map onto an entire side of a zone. It determines the corners, node coordinates and the number of nodes on the side that is matched onto.



### **CKOVER**

CKOVER is called by subroutine MAZONE. It checks to see if the user is trying to write over an existing corner.

### **CKSAME**

CKSAME is called by subroutine MAZONE. It checks all of the corners in a zone to see if the value of a corner passed is that of the other corner.

### **CKNODES**

CKNODES is called by subroutine MAZONE. It checks to see whether the number of nodes on the side being matched are the same as the number of nodes on the other sides of a three-sided zone that have also been matched, or the same as the number of nodes on the opposite side of a four-sided zone that has also been matched.

### **DISNO**

DISNO is called by subroutine MAZONE and CHINDEL. It displays the nodes on a given side of a given zone.

### **CHINDEL**

CHINDEL is called by subroutine ZONE. It allows the user to change, insert and delete individual nodes; it also allows the user to match nodes from any side of any zone onto a side of the zone that is being constructed at the time that subroutine CHINDEL is called.

### **SPEC**

SPEC is called by subroutine ZONE. It solicits the edge coordinates to be specified interactively by the user. Because the order in which the points are listed is important (so that u, v or w directions are consistent), SPEC tells the user the corner to which the first node specified must be the closest.

### **LINR**

LINR is called by subroutines ZONE and CALSIN. It determines the coordinates of the nodes on an edge by interpolating along a straight line between the corners. The user has the option to choose several node spacings: 1) nodes evenly spaced, 2) node space increasing as the square root of the distance, 3) nodes spaced in a linearly increasing fashion, 4) nodes spaced in a linearly decreasing fashion, 5) a more severe case of 3, 6) a more severe case of 4.

### **CIRC**

Subroutine ZONE calls this subroutine to calculate the coordinates of nodes on an edge so that the nodes are spaced evenly along the arc of a circle. Three points determine a circle; this subroutine uses the corners of the edge as two of the points, and asks the user to specify any other point on the circle.

### **QUA**

Subroutines CIRC and CALSIN call this subroutine. It calculates the angle (0 to 2PI) between the positive x-axis, and a line connecting two points.

**CALSIN**

Subroutine ZONE calls this subroutine to calculate the coordinates of nodes on an edge with various spacing options along a sine wave. The sine wave's wavelength, amplitude and starting point are needed.

**SINLINR**

This subroutine is called by CALSIN. It performs a linear interpolation between two points on a sine wave with various spacing options.

**WHICH**

WHICH is called by subroutines SPEC, LINR, CIRC, CALSIN, ZONE, MAZONE, and CHINDEL. It is a short subroutine that, given the side number and the total number of sides in the problem, returns the corner numbers that go with the side under consideration.

**ROTWSIG**

ROTWSIG is called by subroutine MESHPlot. It rounds any real number to two significant digits. It is used for the purpose of rounding the increment on the axes to two significant digits.

**Function ROUNDTASET**

Is called by subroutine MESHPlot. This function rounds off any number to a specified magnitude. It is used to round off the x origin and the y origin to the least significant digit in the axes increment.

## APPENDIX B: EXAMPLE RUN USING PROGRAM MESH

In this example, the items marked with a star are input by the user.

OK, SEG #MESH

Please enter the Name of the next Mesh.  
★SAMPLE

User please note:

CRREL Report # - documents the techniques used in this mesh generator and also contains a user guide for this program. The user guide explains the terminology used in this program and provides some examples. It is strongly suggested that you review at least the user guide section of the report before you proceed.

★0 Please enter 0 to cont.

MESH MAIN MENU  
====

1. Specify # of zones in the mesh. Currently not yet defined.
2. Input the zones.
3. Plot a mesh.
4. Change the name of the mesh. Currently: "SAMPLE"
5. Get a mesh from a file.
6. Save the information on the edges of the mesh "SAMPLE" (not saved)
7. Reduce the bandwidth of a mesh.
8. Exit to Primos Operating System.

★1 Type Selection # then return.

★1 How many zones will there be?

MESH MAIN MENU  
====

1. Specify # of zones in the mesh. Currently # = 1
2. Input the zones.
3. Plot a mesh.
4. Change the name of the mesh. Currently: "SAMPLE"
5. Get a mesh from a file.
6. Save the information on the edges of the mesh "SAMPLE" (not saved)
7. Reduce the bandwidth of a mesh.
8. Exit to Primos Operating System.

★2 Type Selection # then return.

ZONE MENU Current zone # 1  
====

1. Specify the # of sides. The # of sides = not yet defined.
2. Input the corners.
3. Input the # of nodes. The # of nodes = not yet defined.
4. Input the option for generating the nodes along any side.
5. Set the material of the zone. The material # = not yet defined.
6. Set the element type. The element type is not yet defined.
7. Help on the zone construction.
8. Change, insert, delete, or match nodes.
9. Recalculate the edge nodes with previous side options.
10. File Zone # 1 and go to the next Zone or the Main Menu.
11. Emergency exit to the Main Menu.

★1 Type Selection # then return.

★4 How many sides are there in this zone

★0 Specify x and y coordinates of corner # 1

★4 Specify x and y coordinates of corner # 2

★4 Specify x and y coordinates of corner # 3

★0 Specify x and y coordinates of corner # 4

★5 Specify the number of nodes, not counting corners, that you want on side 1

★7 Specify the number of nodes, not counting corners, that you want on side 2

ZONE MENU Current zone # 1

====

1. Specify the # of sides. The # of sides = 4.
2. Change the corners.

	X	Y
Corner 1 is	0.00000	0.00000
Corner 2 is	4.00000	0.00000
Corner 3 is	4.00000	1.50000
Corner 4 is	0.00000	2.00000
3. Change the # of nodes.  
There are 5 nodes on side 1, and 7 nodes on side 2, currently.
4. Input the option for generating the nodes along any side.
5. Set the material of the zone. The material # = not yet defined.
6. Set the element type. The element type is not yet defined.
7. Help on the zone construction.
8. Change, insert, delete, or match nodes.
9. Recalculate the edge nodes with previous side options.
10. File Zone # 1 and go to the next Zone or the Main Menu.
11. Emergency exit to the Main Menu.

★4 Type Selection # then return.

★1 Please enter the side # to specify a node option.

The first node is closest to corner# 1 ( 0.00000, 0.00000)  
The last node is closest to corner# 4 ( 0.00000, 2.00000)

Node specification options for side # 1

- options -----
1. You specify the nodes
  2. Interpolation along a straight line
  3. Interpolation along a circle
  4. Interpolation along a sine wave.

★2 Please specify the option number.

Linr spacing options.

- options -----
1. Evenly spaced nodes.
  2. Spacing as the square root of distance.
  3. Linearly increasing spacing.
  4. Linearly decreasing spacing.
  5. A more severe case 3.
  6. A more severe case 4.

★4 Type selection # and return.

Node # 1	coordinates x =	0.0000	y =	0.5714
Node # 2	coordinates x =	0.0000	y =	1.0476
Node # 3	coordinates x =	0.0000	y =	1.4286
Node # 4	coordinates x =	0.0000	y =	1.7143
Node # 5	coordinates x =	0.0000	y =	1.9048

If all the sides are to have the same option enter  
y or Y otherwise enter anything else.

★N Type 0 then return to continue.  
★0

ZONE MENU Current zone # 1

====

1. Specify the # of sides. The # of sides = 4.
2. Change the corners.

	X	Y
Corner 1 is	0.00000	0.00000
Corner 2 is	4.00000	0.00000
Corner 3 is	4.00000	1.50000
Corner 4 is	0.00000	2.00000
3. Change the # of nodes.  
There are 5 nodes on side 1, and 7 nodes on side 2, currently.

4. Input the option for generating the nodes along any side.
5. Set the material of the zone. The material # = not yet defined.
6. Set the element type. The element type is not yet defined.
7. Help on the zone construction.
8. Change, insert, delete, or match nodes.
9. Recalculate the edge nodes with previous side options.
10. File Zone # 1 and go to the next Zone or the Main Menu.
11. Emergency exit to the Main Menu.

Type Selection # then return.

- ★4 Please enter the side # to specify a node option.  
★2

The first node is closest to corner# 1 ( 0.00000, 0.00000)  
The last node is closest to corner# 2 ( 4.00000, 0.00000)

Node specification options for side # 2

1. You specify the nodes
2. Interpolation along a straight line
3. Interpolation along a circle
4. Interpolation along a sine wave.

Please specify the option number.

- ★3 Specify x and y coords of another point on the circle.  
★2.5

Node # 1	coordinates x =	0.4732	y =	0.2163
Node # 2	coordinates x =	0.9692	y =	0.3731
Node # 3	coordinates x =	1.4807	y =	0.4682
Node # 4	coordinates x =	2.0000	y =	0.5000
Node # 5	coordinates x =	2.5193	y =	0.4682
Node # 6	coordinates x =	3.0308	y =	0.3731
Node # 7	coordinates x =	3.5268	y =	0.2163

Type 0 then return to continue.

★0

ZONE MENU Current zone # 1  
====

1. Specify the # of sides. The # of sides = 4.
2. Change the corners.
 

	X	Y
Corner 1 is	0.00000	0.00000
Corner 2 is	4.00000	0.00000
Corner 3 is	4.00000	1.50000
Corner 4 is	0.00000	2.00000
3. Change the # of nodes.  
There are 5 nodes on side 1, and 7 nodes on side 2, currently.
4. Input the option for generating the nodes along any side.
5. Set the material of the zone. The material # = not yet defined.
6. Set the element type. The element type is not yet defined.
7. Help on the zone construction.
8. Change, insert, delete, or match nodes.
9. Recalculate the edge nodes with previous side options.
10. File Zone # 1 and go to the next Zone or the Main Menu.
11. Emergency exit to the Main Menu.

Type Selection # then return.

- ★4 Please enter the side # to specify a node option.  
★3

The first node is closest to corner# 2 ( 4.00000, 0.00000)  
The last node is closest to corner# 3 ( 4.00000, 1.50000)

Node specification options for side # 3

1. You specify the nodes
2. Interpolation along a straight line
3. Interpolation along a circle
4. Interpolation along a sine wave.

Please specify the option number.

★2

Line spacing options.

1. Evenly spaced nodes.
2. Spacing as the square root of distance.
3. Linearly increasing spacing.
4. Linearly decreasing spacing.
5. A more severe case 3.
6. A more severe case 4.

Type selection # and return.

★4	Node # 1	coordinates x =	4.0000	y =	0.4286
	Node # 2	coordinates x =	4.0000	y =	0.7857
	Node # 3	coordinates x =	4.0000	y =	1.0714
	Node # 4	coordinates x =	4.0000	y =	1.2857
	Node # 5	coordinates x =	4.0000	y =	1.4286

Type 0 then return to continue.

★0

ZONE MENU Current zone # 1  
====

1. Specify the # of sides. The # of sides = 4.
2. Change the corners.
 

	X	Y
Corner 1 is	0.00000	0.00000
Corner 2 is	4.00000	0.00000
Corner 3 is	4.00000	1.50000
Corner 4 is	0.00000	2.00000
3. Change the # of nodes.  
There are 5 nodes on side 1, and 7 nodes on side 2, currently.
4. Input the option for generating the nodes along any side.
5. Set the material of the zone. The material # = not yet defined.
6. Set the element type. The element type is not yet defined.
7. Help on the zone construction.
8. Change, insert, delete, or match nodes.
9. Recalculate the edge nodes with previous side options.
10. File Zone # 1 and go to the next Zone or the Main Menu.
11. Emergency exit to the Main Menu.

Type Selection # then return.

- ★4 Please enter the side # to specify a node option.  
★4

The first node is closest to corner# 4 ( 0.00000, 2.00000)  
The last node is closest to corner# 3 ( 4.00000, 1.50000)

Node specification options for side # 4  
-----

1. You specify the nodes
2. Interpolation along a straight line
3. Interpolation along a circle
4. Interpolation along a sine wave.

Please specify the option number.

- ★4 Please enter the desired wave length of the sine wave.  
(1=PI 2=2\*PI .5=PI/2 ETC.)

- ★1.5 Please enter the starting point of this sin wave.  
(1=PI 2=2\*PI .5=PI/2 ETC.)

- ★0 Please enter the amplitude of the sin wave.

★1.

Sin spacing options.  
-----

1. Evenly spaced nodes.
2. Spacing as the square root of distance.
3. Linearly increasing spacing.
4. Linearly decreasing spacing.
5. A more severe case 3.
6. A more severe case 4.

Type selection # and return.

- ★1
- |          |                 |        |     |        |
|----------|-----------------|--------|-----|--------|
| Node # 1 | coordinates x = | 0.5844 | y = | 2.6128 |
| Node # 2 | coordinates x = | 1.1456 | y = | 3.0398 |
| Node # 3 | coordinates x = | 1.6682 | y = | 3.1578 |
| Node # 4 | coordinates x = | 2.1497 | y = | 2.9478 |
| Node # 5 | coordinates x = | 2.6017 | y = | 2.5013 |
| Node # 6 | coordinates x = | 3.0456 | y = | 1.9895 |
| Node # 7 | coordinates x = | 3.5054 | y = | 1.6057 |
- Type 0 then return to continue.

★0

ZONE MENU Current zone # 1  
====

1. Specify the # of sides. The # of sides = 4.
2. Change the corners.
 

	X	Y
Corner 1 is	0.00000	0.00000
Corner 2 is	4.00000	0.00000
Corner 3 is	4.00000	1.50000
Corner 4 is	0.00000	2.00000
3. Change the # of nodes.  
There are 5 nodes on side 1, and 7 nodes on side 2, currently.
4. Change the option for generating the nodes along any side.
5. Set the material of the zone. The material # = not yet defined.
6. Set the element type. The element type is not yet defined.
7. Help on the zone construction.
8. Change, insert, delete, or match nodes.
9. Recalculate the edge nodes with previous side options.
10. File Zone # 1 and go to the next Zone or the Main Menu.
11. Emergency exit to the Main Menu.

Type Selection # then return.

- ★10 You still must select #5 to specify the material type for this zone.  
Type 0 then return to continue.  
★0

ZONE MENU Current zone # 1  
====

1. Specify the # of sides. The # of sides = 4.
2. Change the corners. X Y  
 Corner 1 is 0.00000 0.00000  
 Corner 2 is 4.00000 0.00000  
 Corner 3 is 4.00000 1.50000  
 Corner 4 is 0.00000 2.00000
3. Change the # of nodes.  
 There are 5 nodes on side 1, and 7 nodes on side 2, currently.
4. Change the option for generating the nodes along any side.
5. Set the material of the zone. The material # = not yet defined.
6. Set the element type. The element type is not yet defined.
7. Help on the zone construction.
8. Change, insert, delete, or match nodes.
9. Recalculate the edge nodes with previous side options.
10. File Zone # 1 and go to the next Zone or the Main Menu.
11. Emergency exit to the Main Menu.

Type Selection # then return.

★5

Currently the material type for zone # 1 not yet defined.

The integer associated with a material may be used to index the material types later, in the finite element program. The main use of the material type in MESH is to indicate the color number of the pen to be used for plotting. The different materials in your problem should be numbered from 1 to N. Each zone must be assigned a material type.

Please type in the material type for zone # 1:

★1

ZONE MENU Current zone # 1  
====

1. Specify the # of sides. The # of sides = 4.
2. Change the corners. X Y  
 Corner 1 is 0.00000 0.00000  
 Corner 2 is 4.00000 0.00000  
 Corner 3 is 4.00000 1.50000  
 Corner 4 is 0.00000 2.00000
3. Change the # of nodes.  
 There are 5 nodes on side 1, and 7 nodes on side 2, currently.
4. Change the option for generating the nodes along any side.
5. Change the material of the zone. The material # = 1.
6. Set the element type. The element type is not yet defined.
7. Help on the zone construction.
8. Change, insert, delete, or match nodes.
9. Recalculate the edge nodes with previous side options.
10. File Zone # 1 and go to the next Zone or the Main Menu.
11. Emergency exit to the Main Menu.

Type Selection # then return.

★6

Element type options for zone # 1  
-----

1. Triangular elements.
2. Four noded Quadrilateral elements.
3. Eight noded Quadrilateral elements.

Please specify the option number.

★1

ZONE MENU Current zone # 1  
====

1. Specify the # of sides. The # of sides = 4.
2. Change the corners. X Y  
 Corner 1 is 0.00000 0.00000  
 Corner 2 is 4.00000 0.00000  
 Corner 3 is 4.00000 1.50000  
 Corner 4 is 0.00000 2.00000
3. Change the # of nodes.  
 There are 5 nodes on side 1, and 7 nodes on side 2, currently.
4. Change the option for generating the nodes along any side.
5. Change the material of the zone. The material # = 1.
6. Change the element type. The element type is triangular.
7. Help on the zone construction.
8. Change, insert, delete, or match nodes.
9. Recalculate the edge nodes with previous side options.
10. File Zone # 1 and go to the next Zone or the Main Menu.
11. Emergency exit to the Main Menu.

Type Selection # then return.

★10

The half bandwidth of the mesh is 8.

Please enter 0 to cont.

★0

MESH MAIN MENU  
====

1. Specify # of zones in the mesh. Currently # = 1
2. Input the zones.
3. Plot a mesh.
4. Change the name of the mesh. Currently: "SAMPLE "
5. Get a mesh from a file.
6. Save the information on the edges of the mesh "SAMPLE " (not saved)
7. Reduce the bandwidth of a mesh.
8. Exit to Primos Operating System.

★3 Type Selection # then return.

Are you assigned to the HP plotter? If not you must exit to Primos, and then use ASHP to assign the HP plotter. Please enter n or N to exit and anything else to continue.

Would you like to plot the current mesh?  
(y or Y = yes and anything else = no)

★Y

Note please load the plotter with paper before proceeding.

Please enter "1" if you want the nodes numbered and anything else otherwise.

★N

Use ASHP to assign AMLC 7

00 YOU WISH TO PLOT THIS TIME

★Y

Do you want the mesh titled? Enter Y or y for yes anything else for no.

★Y

Please enter the label for the mesh.

★SAMPLE

Do you want the axes labeled? Enter Y or y for yes anything else for no.

★Y

Please enter the label for the x-axis.

★X

Please enter the label for the y-axis.

★Y

Do you wish to have all the zones plotted in black (pen color #1) regardless of the material type? (y or Y = yes and anything else =

★Y

Please enter the number of inches to be on the X-axis.

★6

Please enter the number of inches to be on the Y-axis.

★6

MESH MAIN MENU  
====

1. Specify # of zones in the mesh. Currently # = 1
2. Input the zones.
3. Plot a mesh.
4. Change the name of the mesh. Currently: "SAMPLE "
5. Get a mesh from a file.
6. Save the information on the edges of the mesh "SAMPLE " (not saved)
7. Reduce the bandwidth of a mesh.
8. Exit to Primos Operating System.

★6 Type Selection # then return.

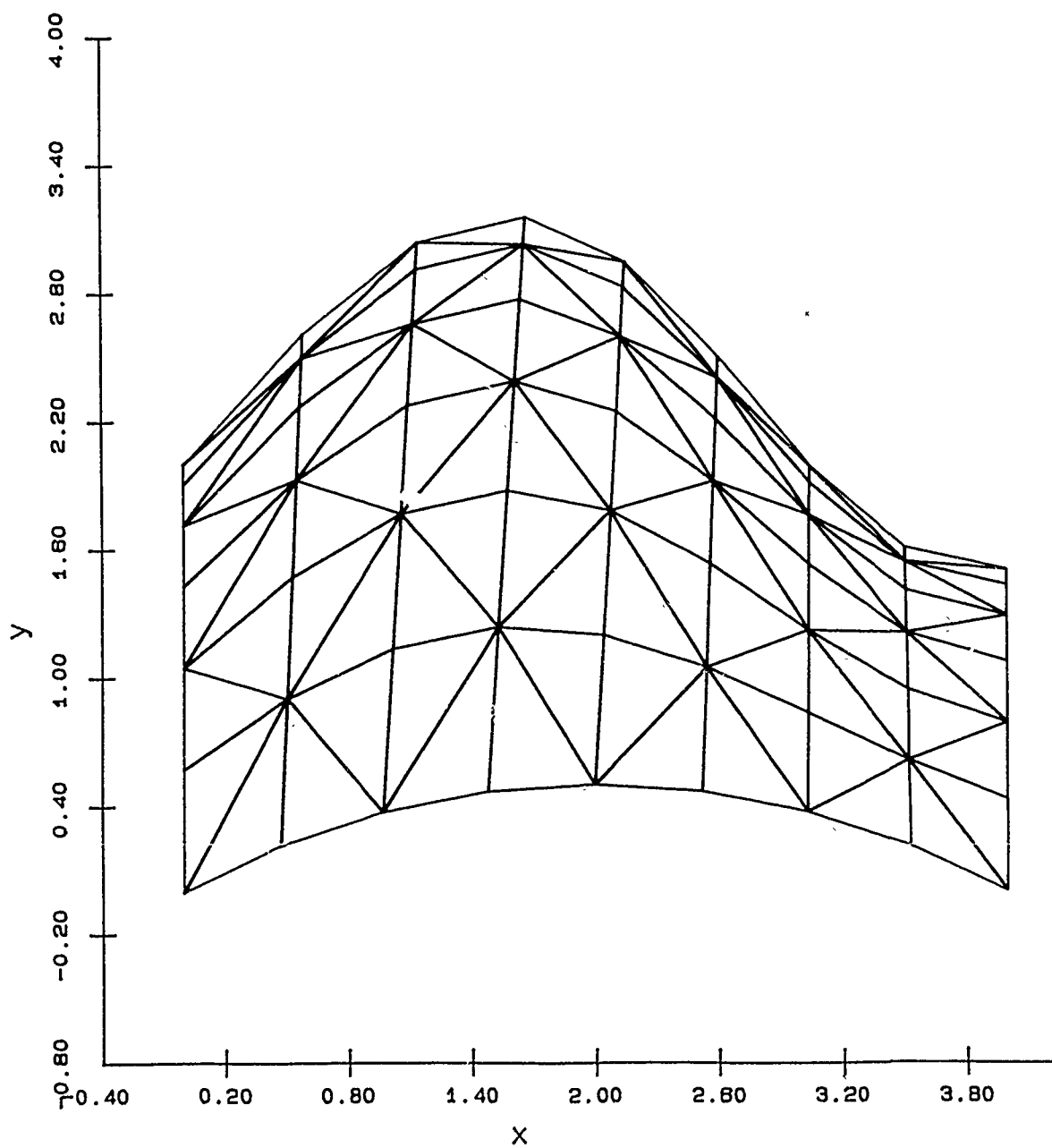
MESH MAIN MENU  
====

1. Specify # of zones in the mesh. Currently # = 1
2. Input the zones.
3. Plot a mesh.
4. Change the name of the mesh. Currently: "SAMPLE "
5. Get a mesh from a file.
6. Save the information on the edges of the mesh "SAMPLE " (saved)
7. Reduce the bandwidth of a mesh.
8. Exit to Primos Operating System.

★8 Type Selection # then return.



# Sample



A facsimile catalog card in Library of Congress MARC format is reproduced below.

Albert, Mary Remley

An automatic finite element mesh generator / by Mary Remley Albert and James L. Warren. Hanover, N.H.: U.S. Army Cold Regions Research and Engineering Laboratory; Springfield, Va.: available from National Technical Information Service, 1987.

iii, 32 p., illus.; 28 cm. (CRREL Report 87-18.)

Bibliography: p. 15.

1. Computer code. 2. Differential equations. 3. Finite element. 4. Fluid dynamics. 5. Heat transfer. 6. Mathematical methods. 7. Solid mechanics. I. Warren, James L. II. United States Army. Corps of Engineers. III. Cold Regions Research and Engineering Laboratory. IV. Series: CRREL Report 87-18.